

**MASSACHUSETTS INSTITUTE OF TECHNOLOGY**

**Analyzing wind flow around the square plate using**

**ADINA**

**2.094 - Project**

Ankur Bajoria

May 1, 2008

## **Acknowledgement**

I would like to thank ADINA R & D, Inc for the full version of the software. I am grateful to Professor K.J. Bathe who has made the difficult topic of finite element easy to understand in his course 2.094. Special thanks to Dr. Shanhong Ji who provided valuable guidance to improve the model and complete the project. Equally important are the inputs from Renard Gamaliel and Ali Lame. Their comments have contributed to this project's improvement.

## Table of Contents

Acknowledgement .....	2
Table of Contents .....	3
List of Figures .....	4
1. Introduction .....	6
2. Modeling and governing parameters in ADINA .....	6
3. Results .....	11
4. Conclusion .....	23
5. References .....	23

## List of Figures

Figure 1: model outline .....	7
Figure 2: model in ADINA.....	8
Figure 3: boundary conditions and loads applied to model.....	8
Figure 4: model mesh.....	9
Figure 5-a: $C_p$ ( $i$ ) and face .....	11
Figure 5-b: $C_p$ plot for three face (viscosity=0.01).....	11
Figure 6-a: $C_p$ plot simulation by S. Murakami [1] .....	12
Figure 6-b: $C_p$ plot simulation and experimental by S. Murakami [1] .....	12
Figure 7: velocity magnitude .....	13
Figure 8: y-velocity.....	13
Figure 9: z-velocity .....	14
Figure 10: Omega-yz.....	14
Figure 11: Nodal pressure .....	14
Figure 12: Turbulence kinetic energy .....	15
Figure 13: Cell Reynolds number .....	15
Figure 14: $C_p$ plot for three face (viscosity=0.001).....	16
Figure 15: velocity magnitude .....	16
Figure 16: y-velocity .....	17
Figure 17: z-velocity .....	17
Figure 18: Omega-yz.....	17
Figure 19: Nodal pressure .....	18

Figure 20: Turbulence kinetic energy .....	18
Figure 21: Cell Reynolds number .....	19
Figure 22: Cp plot for three face (viscosity=0.0001).....	20
Figure 23: velocity magnitude .....	20
Figure 24: y-velocity .....	21
Figure 25: z-velocity .....	21
Figure 26: Omega-yz.....	21
Figure 27: Nodal pressure .....	22
Figure 28: Turbulence kinetic energy .....	22
Figure 29: Cell Reynolds number .....	22

## 1. Introduction

In this project, I analyze the wind flow around the square plate using k- $\epsilon$  turbulence model. Then I increase the Reynolds number of the problem by decreasing the viscosity from 0.01-0.0001 N.s/m<sup>2</sup> in three different models. This helps to study the effect of introducing non-linearity in the model and various measures that are required for the solution to converge. The flow assumptions are planar, incompressible and steady-state.

## 2. Modeling and governing parameters in ADINA

The model outline is shown in the fig 1. The boundary conditions used and the applied loads are also shown. The model is developed in ADINA with the above given parameters. The wind has been assumed to have the density 1kg/m<sup>3</sup>. The viscosity of wind is varied as 0.01, 0.001 and 0.0001 N.s/m<sup>2</sup> in three different model run so as to study the effect of increasing the Reynolds number.

The model has been meshed using the 4 node elements. The mesh density is fine near the edge of the square plate to capture the effect. The model has 2300 elements and 9200 nodes. The various parameters that are plotted for the comparison are: y-velocity, z-velocity, velocity magnitude, turbulence, omega-yz, turbulence kinetic energy, cell Reynolds number and nodal pressure. The model line plot for the different face of square has been plotted with the user defined variable coefficient of pressure,  $C_p$ . the coefficient of pressure is defined as the ratio of nodal pressure to the input kinetic energy per unit volume.

$$C_p = \frac{\text{Nodal pressure}}{\frac{1}{2}\rho U^2} \quad (1.1)$$

where,

$\rho$  = density

$U$  = inlet velocity

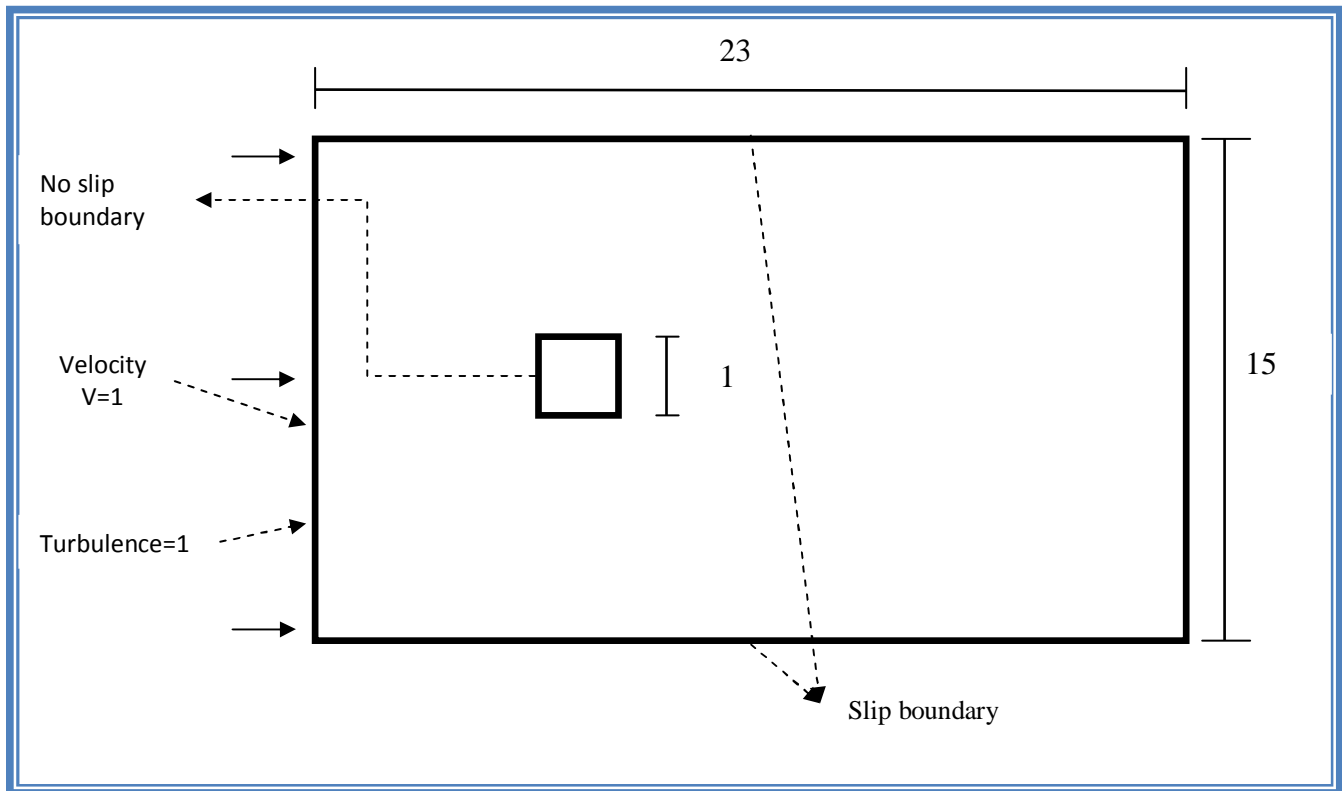


Figure 1: model outline

The results of  $C_p$  are then compared to the experimental and simulation data that has been published to verify the model.

The figure 2 shows the model in ADINA. It has been divided into 8 surfaces for the purpose of desired mesh. The figure 3 shows the boundary conditions applied to the model. Selection of proper boundary conditions is very important for the fluid problem for solution to converge. The special boundary condition of no-slip is applied on all the four lines of the plate as indicated in

the model outline. The special boundary condition of slip is applied to the two sides of the flow boundary. This special boundary condition is different from the normal boundary condition used in ADINA in the sense that these are discretized within boundary elements. The loads applied for the turbulence model is velocity and turbulence load at the inlet boundary. Here a velocity of  $v=1$  and turbulence=1 is applied.

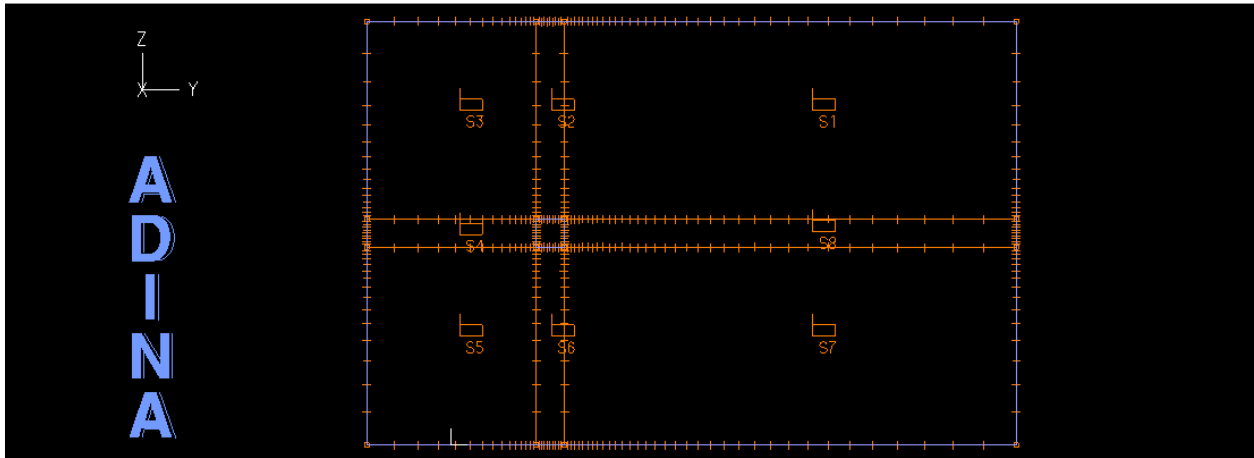


Figure 2: model in ADINA

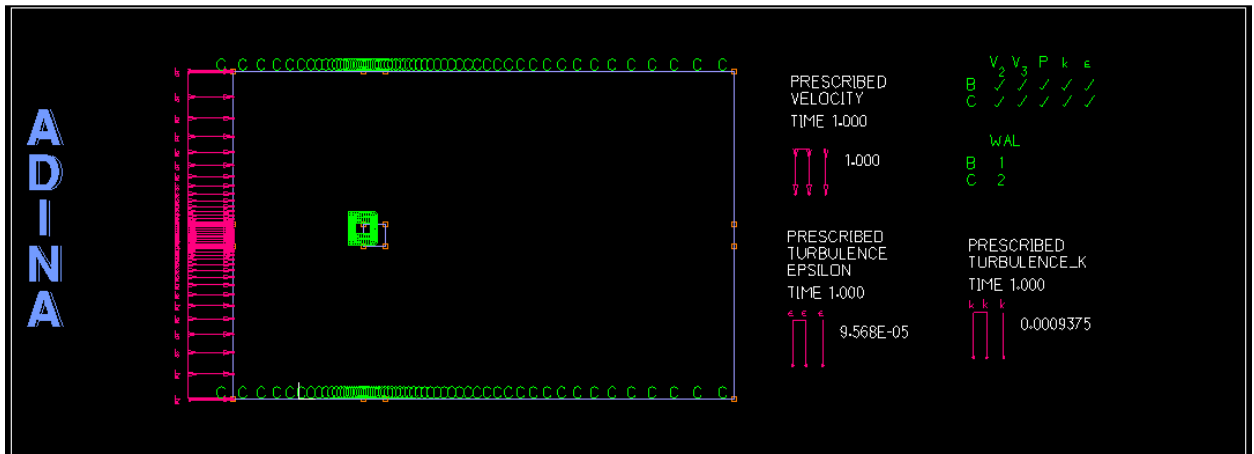


Figure 3: boundary conditions and loads applied to model



The turbulence intensity of 0.025 and dissipation length of 1 is used. The prescribed value of the turbulence and the kinetic energy at the inlet boundary are calculated from the following equations.

$$K = \frac{3}{2}(\bar{iv})^2 \quad (1.2)$$

$$\varepsilon = K^{\frac{3}{2}} / (0.3\bar{L}) \quad (1.3)$$

where  $\bar{v}$ ,  $\bar{L}$  and  $\bar{i}$  are velocity scale, length scale and turbulence intensity respectively. The zero turbulence flux is implicitly assumed at the boundary where no turbulence is explicitly specified

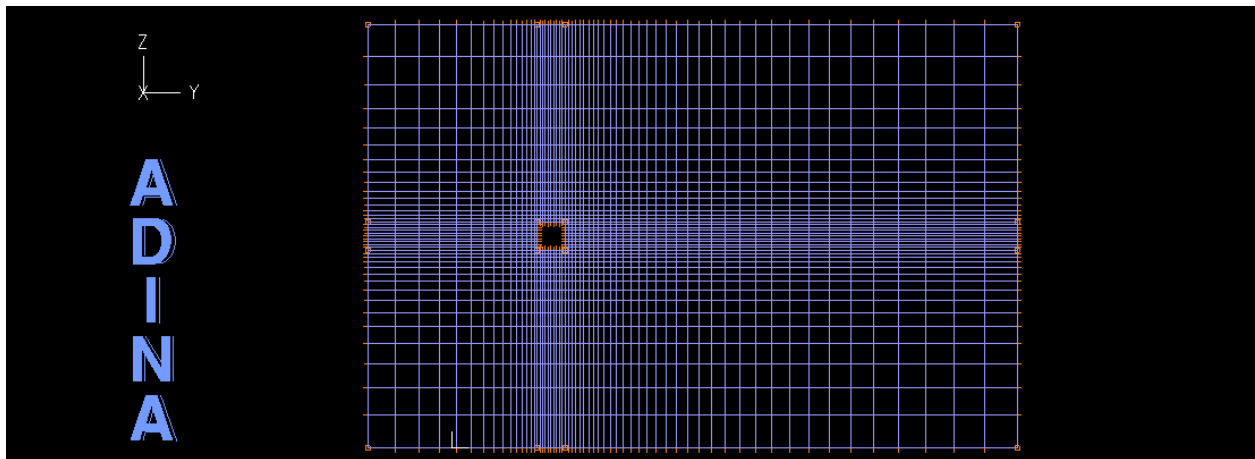


Figure 4: model mesh

The figure 4 shows the mesh used for the analysis. Finer mesh is used near the plate boundary. All the surfaces have the ratio 1 for the largest to smallest element. 4 node quadrilateral elements are used in which velocity is computed at all 4 nodes and pressure computed at the centre. The default values for the K-  $\epsilon$  turbulence material model for the various parameters are used. These values are given below:

$$\begin{array}{lll} C_{\mu}=0.09, & C_1=1.42, & C_2=1.92, \\ C_3=0.8, & \sigma_k=1, & \sigma_{\epsilon}=1.3, \\ \sigma_{\epsilon}=0.9, & d_w^+=70, & \kappa_0=0.4 \end{array}$$

The automatic time stepping function (ATS) with 10 sub-divisions is used for the steady-state incompressible flow analysis. The parameters that have been explained above remains same for all the models. Apart from the above, the parameters that vary in each of the model are now discussed.

In Model 1, [Viscosity=0.01](#), default time function which is full loads applied from the start of analysis, default time step which is full loads at first step and number of iteration in newton method are 15 is used.

In Model 2, [Viscosity=0.001](#), a linear time function which is full loads applied linearly from zero to full load in time one, smaller time step which is full loads applied to the model in two steps of magnitude 0.1 & 0.9 and number of iteration in newton method are 100 is used.

In Model 3, [Viscosity=0.0001](#), a linear time function which is full loads applied linearly from zero to full load in time one, smaller time step which is full loads applied to the model in three steps of magnitude 0.1, 0.3 & 0.6 and number of iteration in newton method are 100 is used.

These changes are required to make the set of equations stable under high Reynolds number which is increased due to the decrease in the viscosity of the fluid. The solution doesn't converge if full loads are applied in one step and hence smaller time step is required to reach convergence.

### 3. Results

To verify the model and the boundary conditions used the results are compared to the experimental and simulation data that has been published for the flow around the square plate.

For the purpose of comparison the graphs of pressure coefficient ( $C_p$ ) for the different face of the plate have been plotted. The figure 5-a shows the face and corresponding  $C_p (i)$ ,  $i=1, 2, 3$ . The figure 5-b shows the graph of  $C_p$  for different face. The figure 6-a shows the corresponding graph for experimental and 6-b simulation data [1].

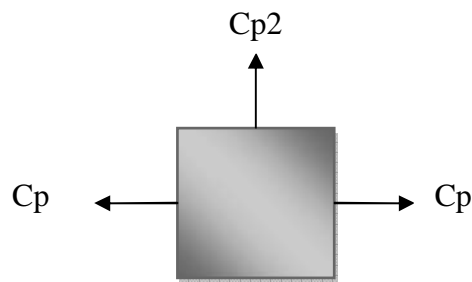


Figure 5-a:  $C_p (i)$  and face

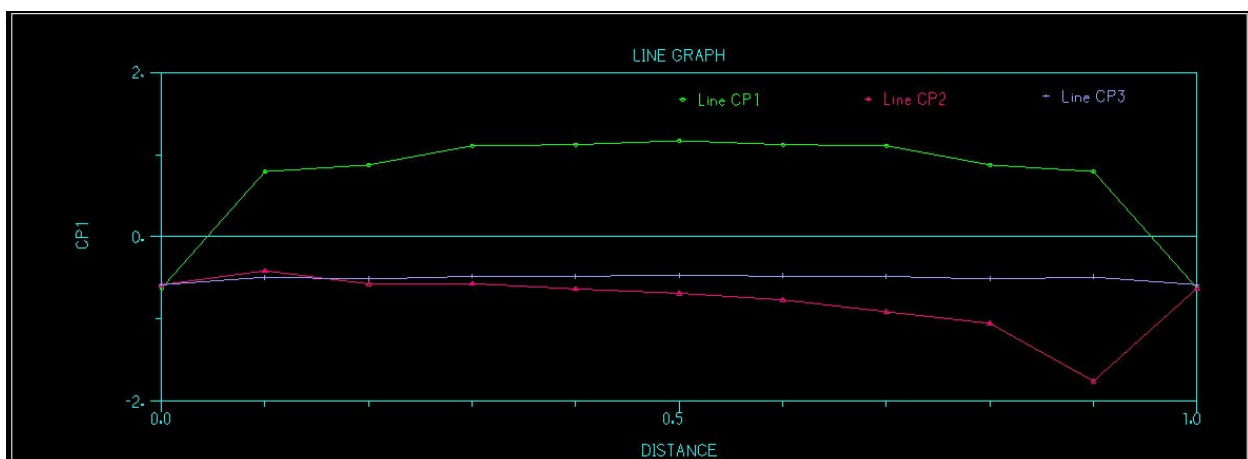


Figure 5-b:  $C_p$  plot for three face (viscosity=0.01)

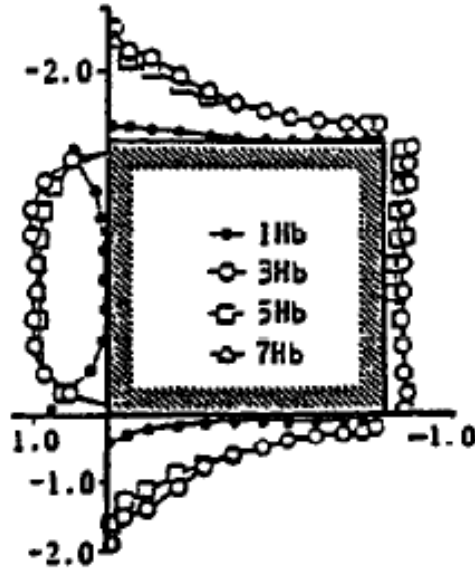


Figure 6-a:  $C_p$  plot simulation by S. Murakami [1]

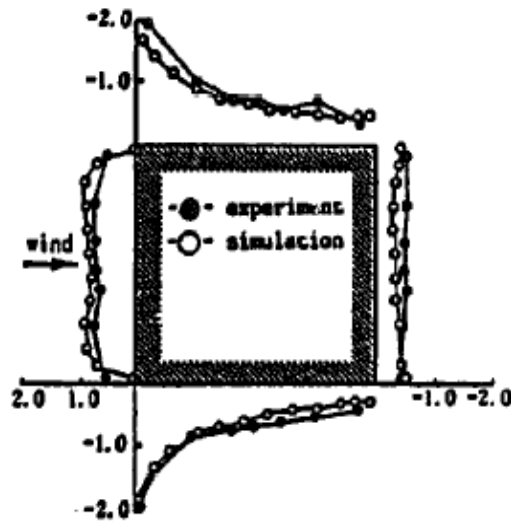


Figure 6-b:  $C_p$  plot simulation and experimental by S. Murakami [1]

Courtesy of Elsevier, Inc., <http://www.sciencedirect.com>. Used with permission.

From the above comparison we see the calculated  $C_p$  from the model is in agreement with the experimental results. The various plots for the first model are shown below:

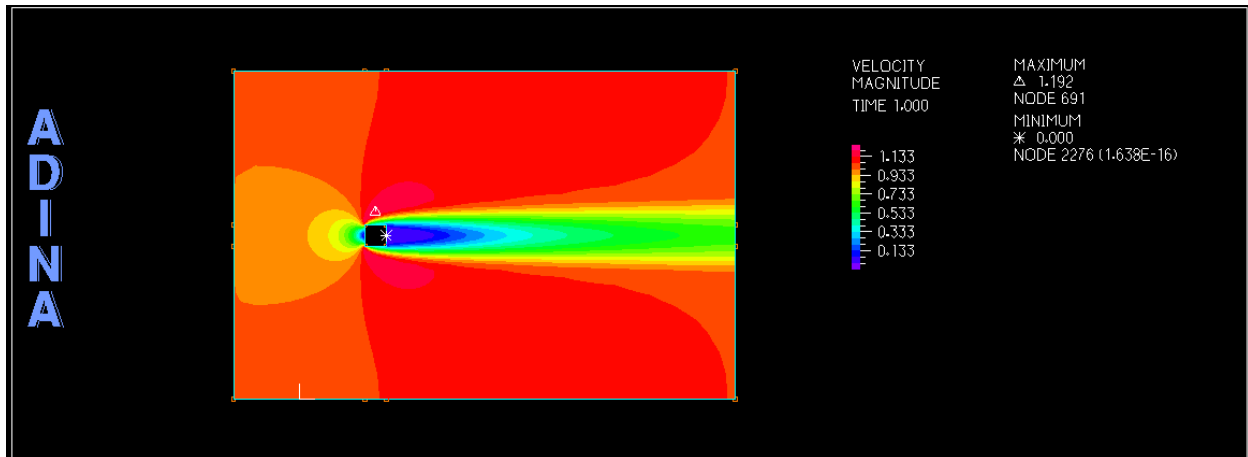


Figure 7: velocity magnitude

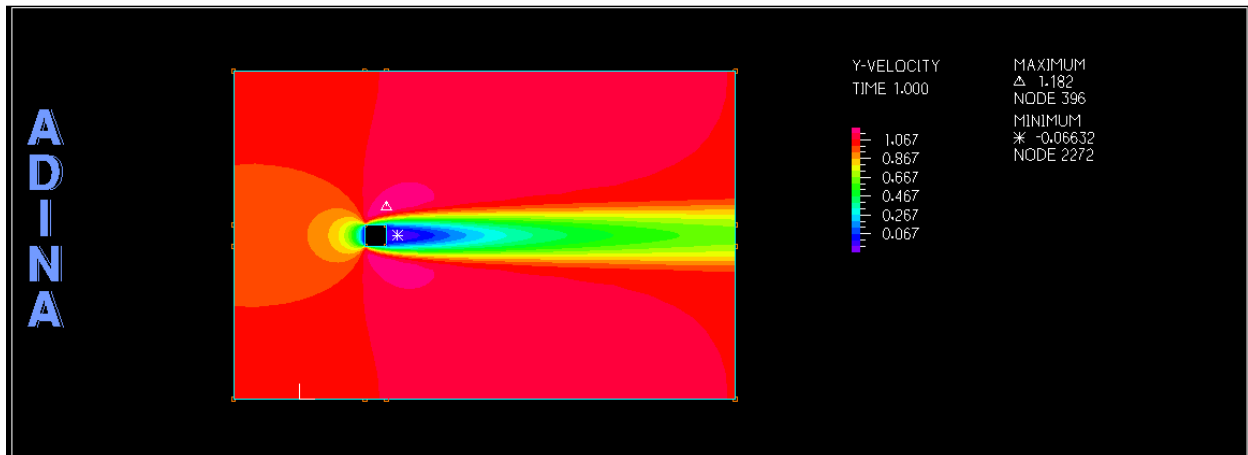


Figure 8: y-velocity

The velocity distribution due to the square in the flow field is shown by figure 7, 8, 9. Zero and negative velocity is developed behind the square, which causes the circulation. From the figure 10, the formation of vortices around the corner of the square plate is indicated.

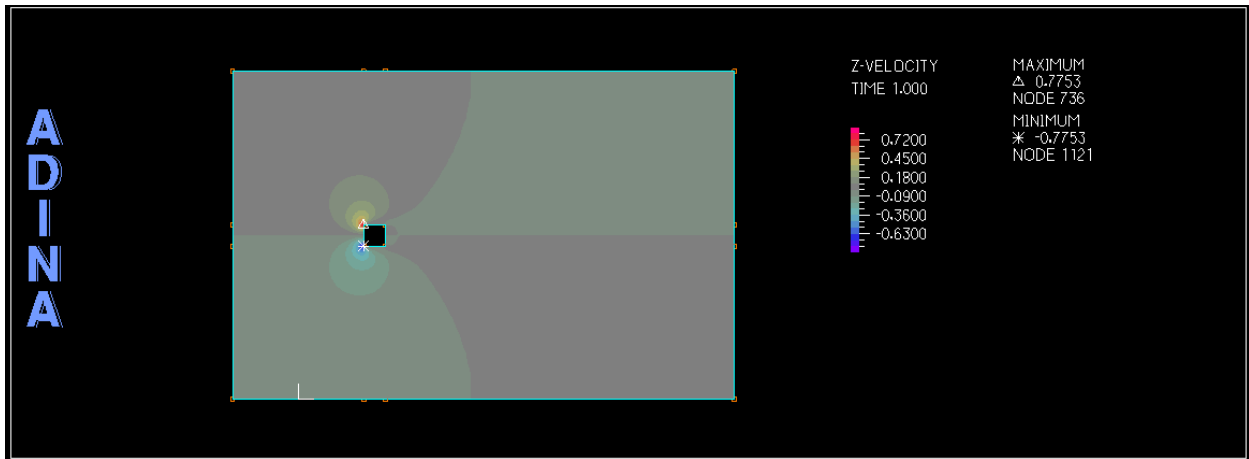


Figure 9: z-velocity

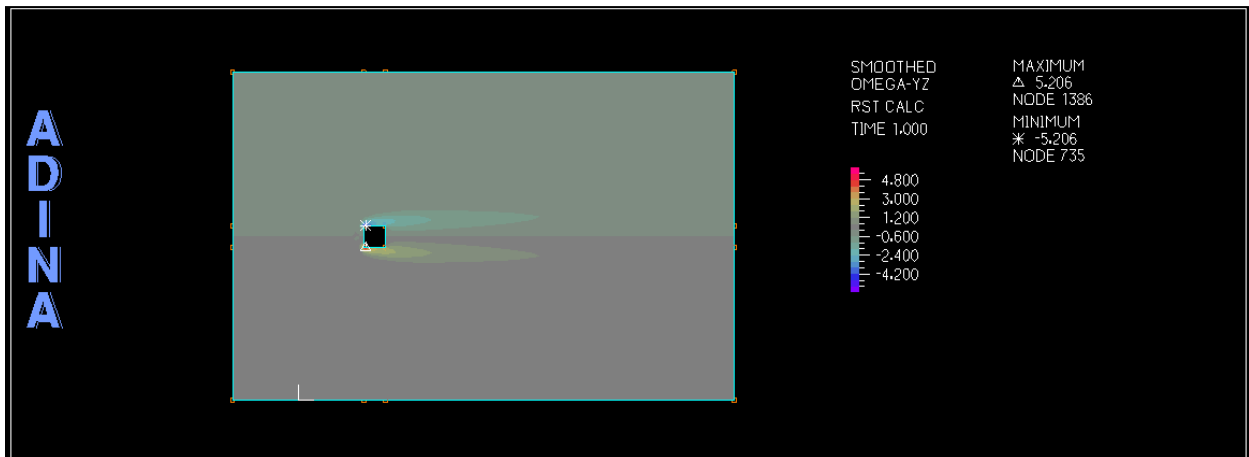


Figure 10: Omega-yz

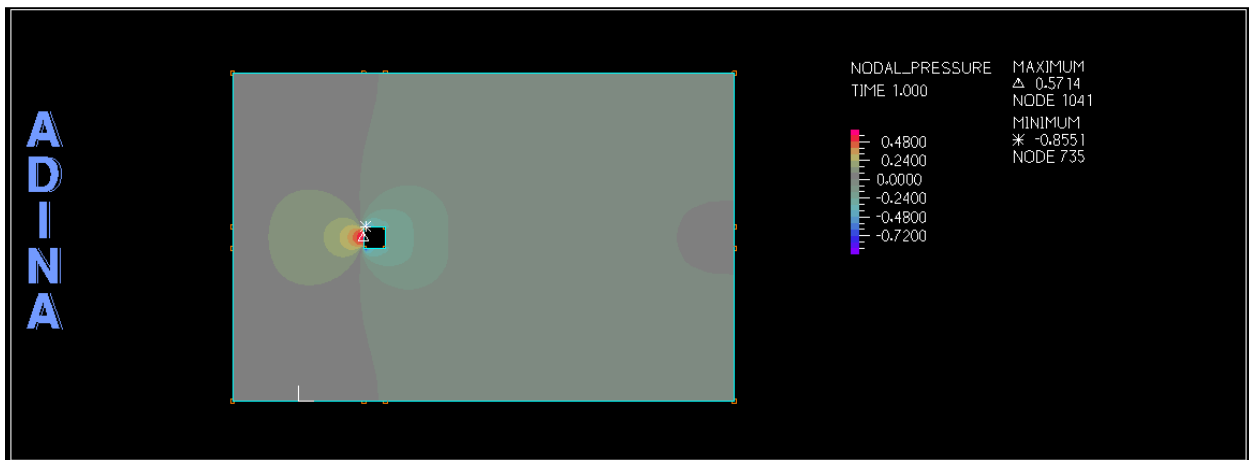


Figure 11: Nodal pressure

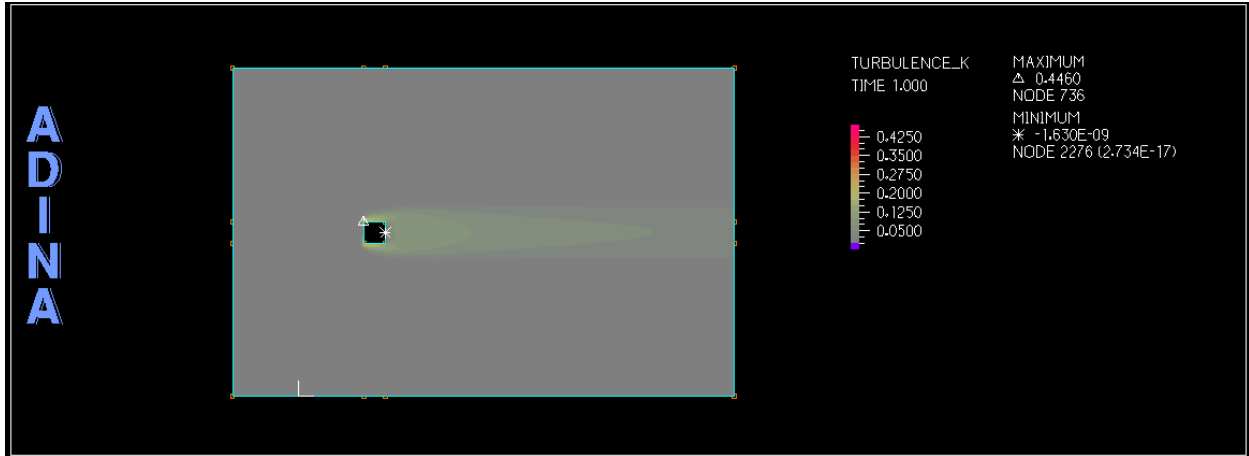


Figure 12: Turbulence kinetic energy

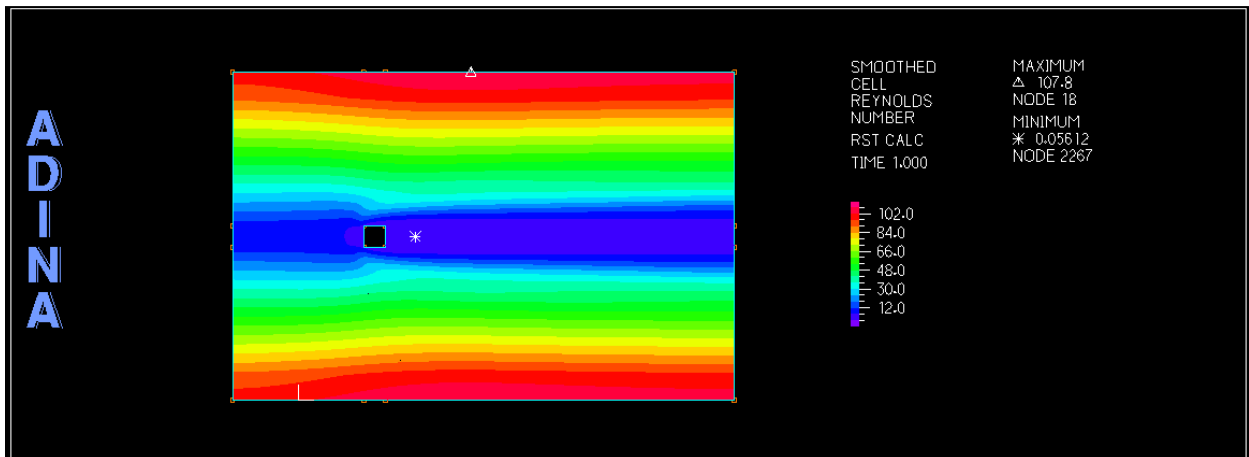


Figure 13: Cell Reynolds number

Plots for model 2(viscosity=0.001) are now shown:

From the figure 15, the increase in the magnitude of velocity as compared to the same plot at lower Reynolds number can be noticed. Also other plots show the increase in the intensity of parameters such as kinetic energy, turbulence and omega-yz etc.

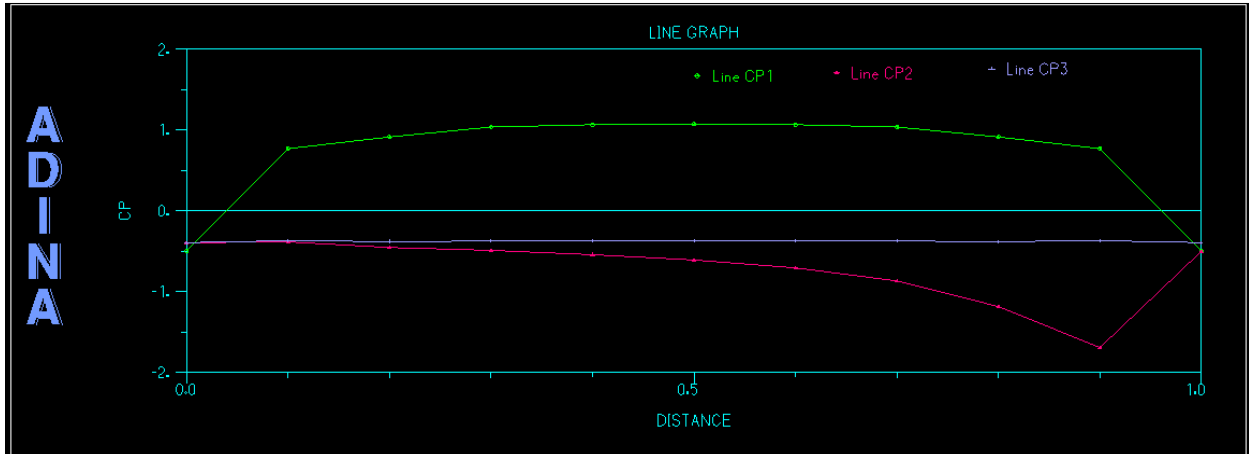


Figure 14: Cp plot for three face (viscosity=0.001)

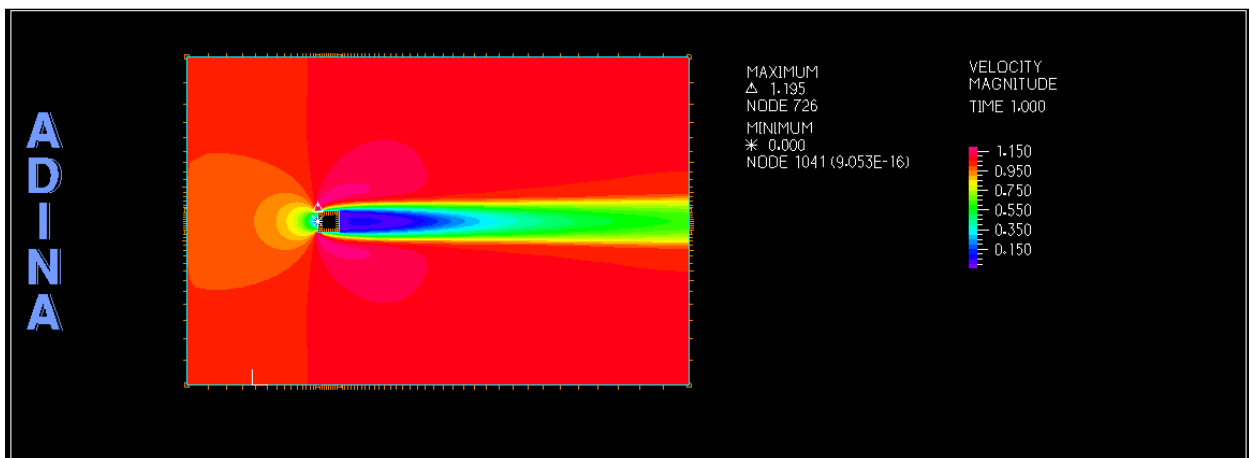


Figure 15: velocity magnitude



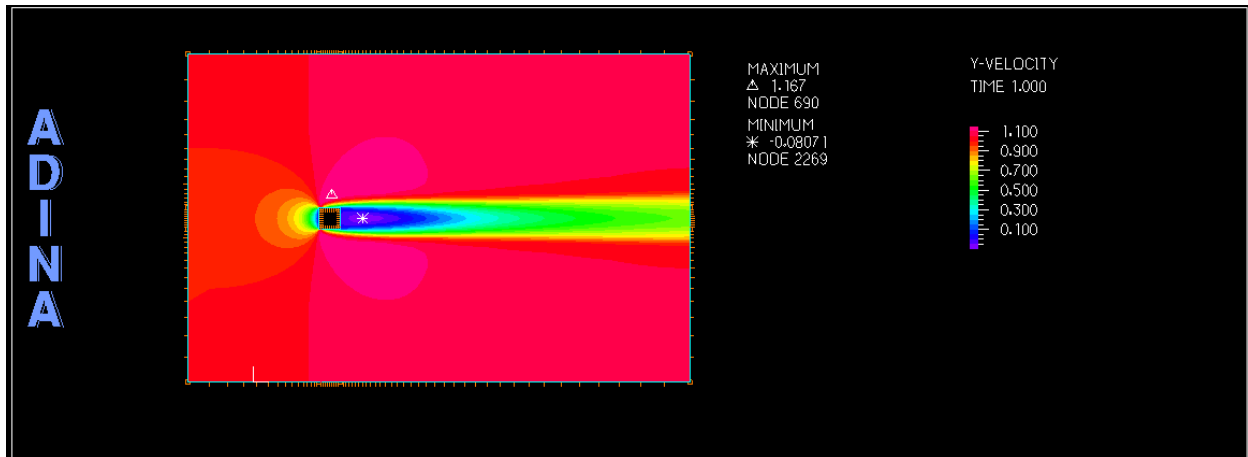


Figure 16: y-velocity

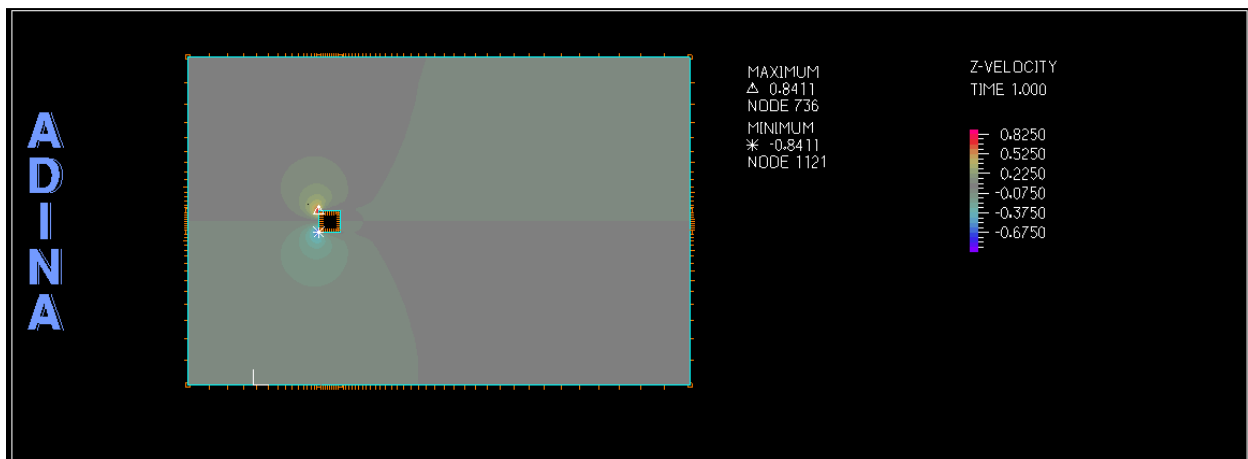


Figure 17: z-velocity

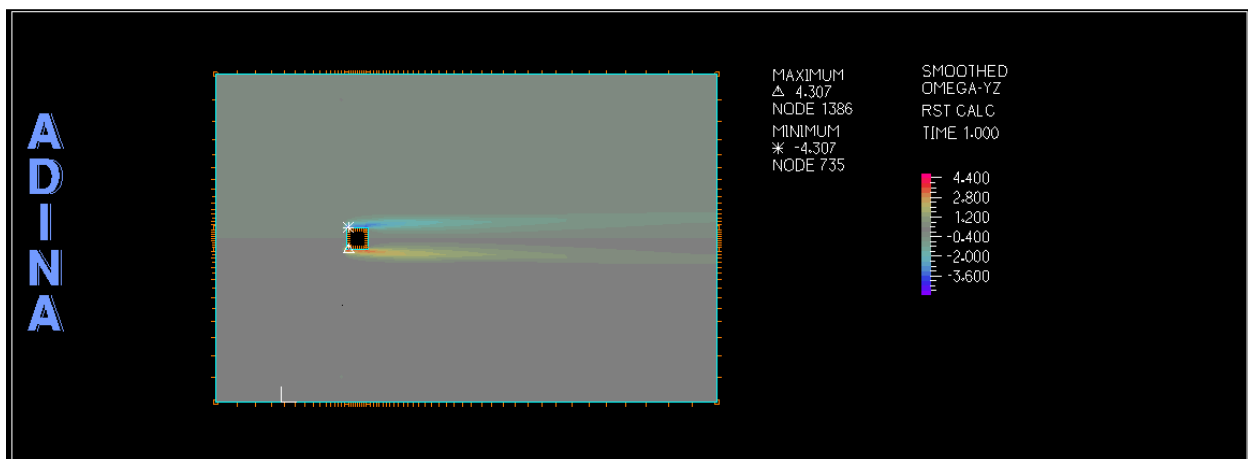


Figure 18: Omega-yz

From figure 18, vortices formation at the edge of the plate is shown. Due to the increase in the Reynolds number of the flow effect of these vortices is increased in the downstream of the flow. The nodal pressure is increased but only by small amount which would be generally negligible when a factor of safety is used in the design of the structure. Figure 20, shows the highly turbulent region behind the plate which indicates the formation of eddies in the leeward region of flow.

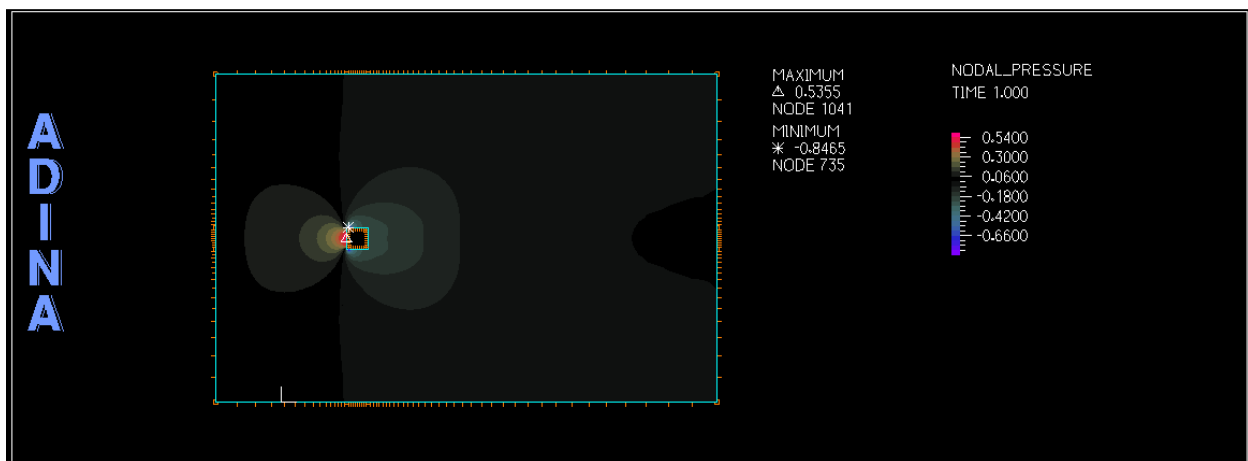


Figure 19: Nodal pressure

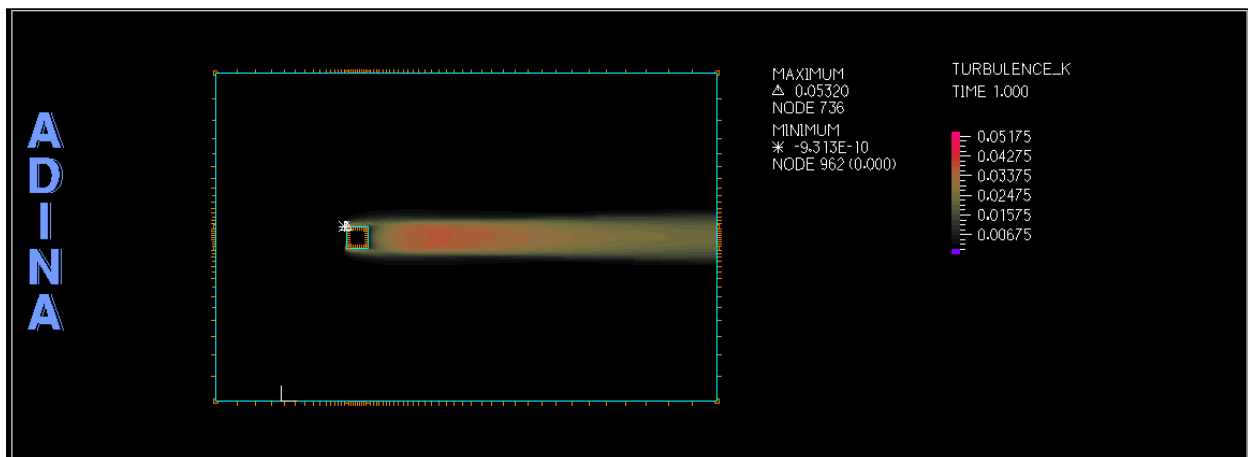


Figure 20: Turbulence kinetic energy

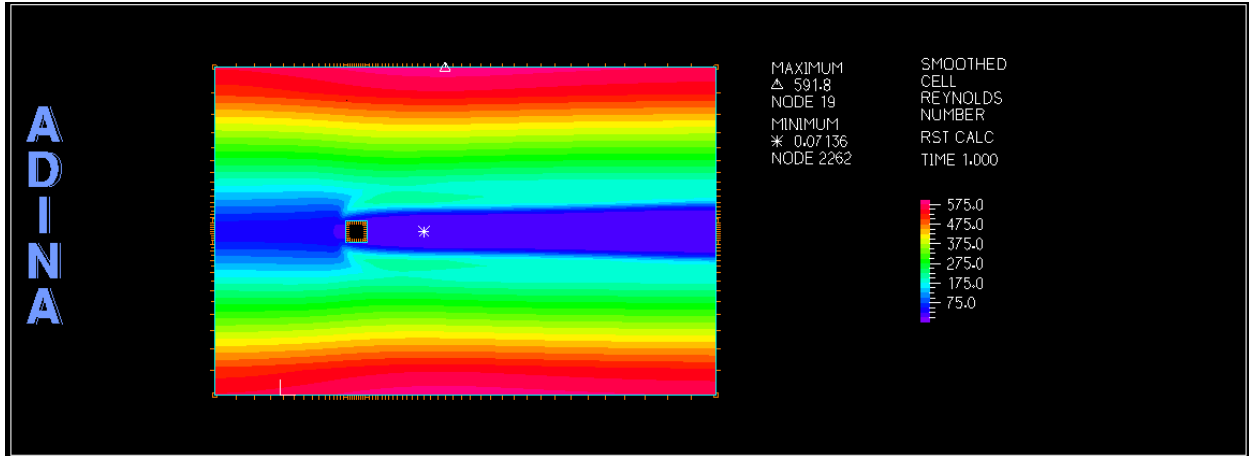


Figure 21: Cell Reynolds number

Plot for model 3 (viscosity=0.0001)

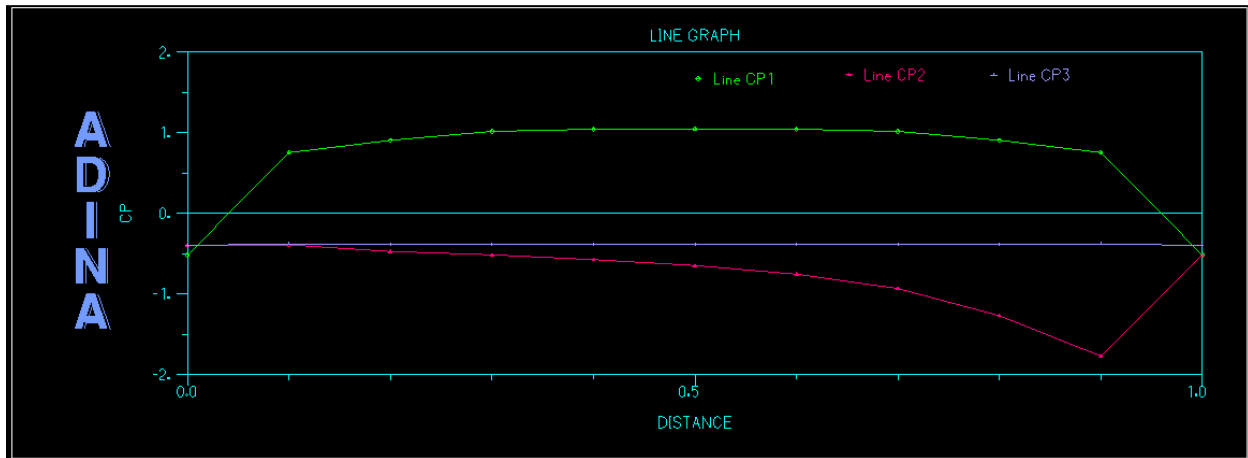


Figure 22: Cp plot for three face (viscosity=0.0001)

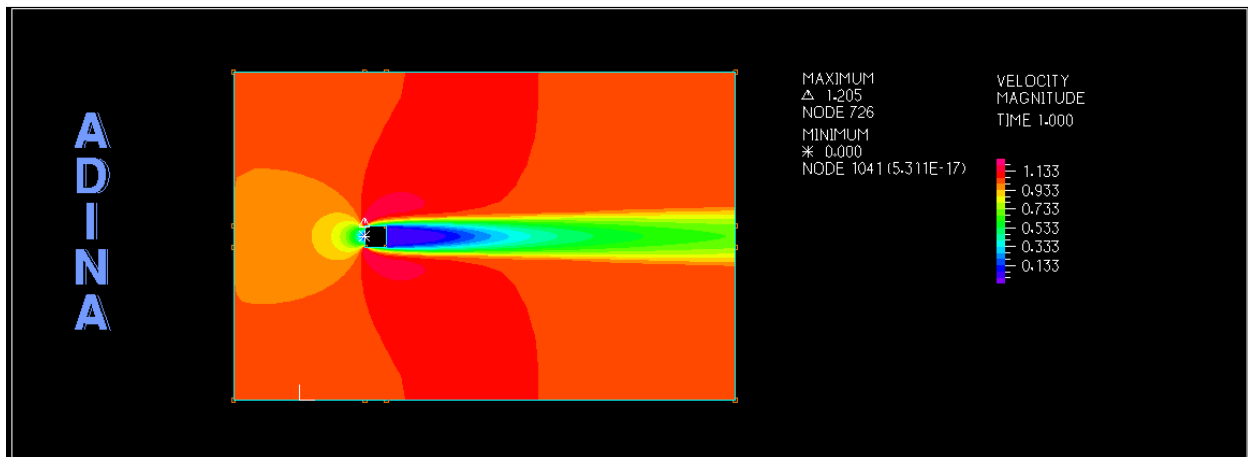


Figure 23: velocity magnitude

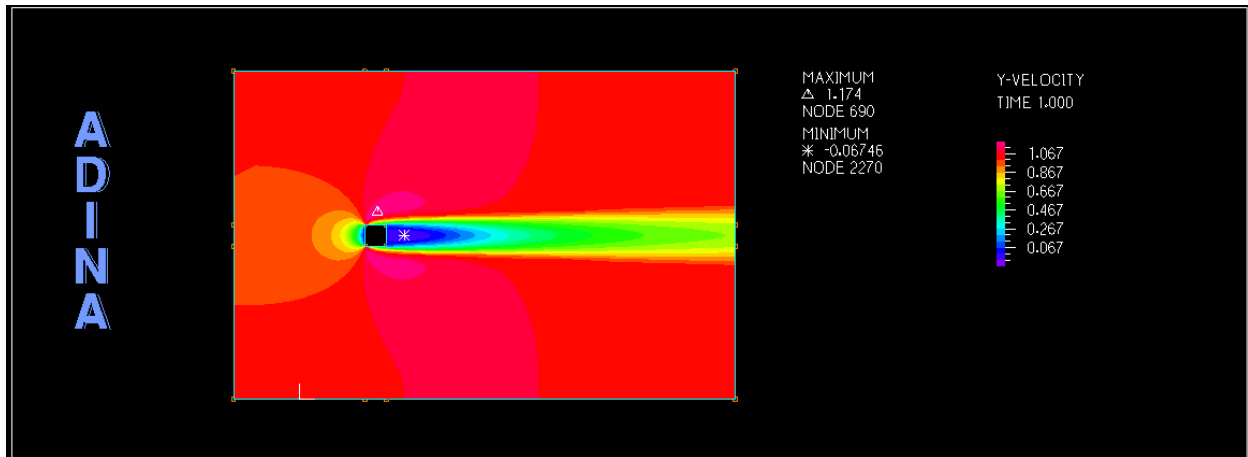


Figure 24: y-velocity

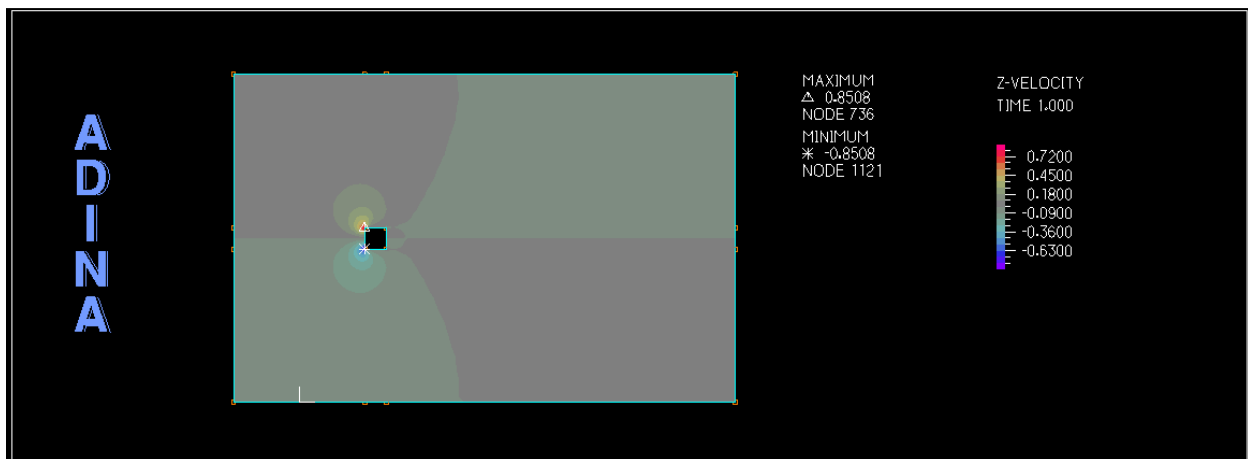


Figure 25: z-velocity

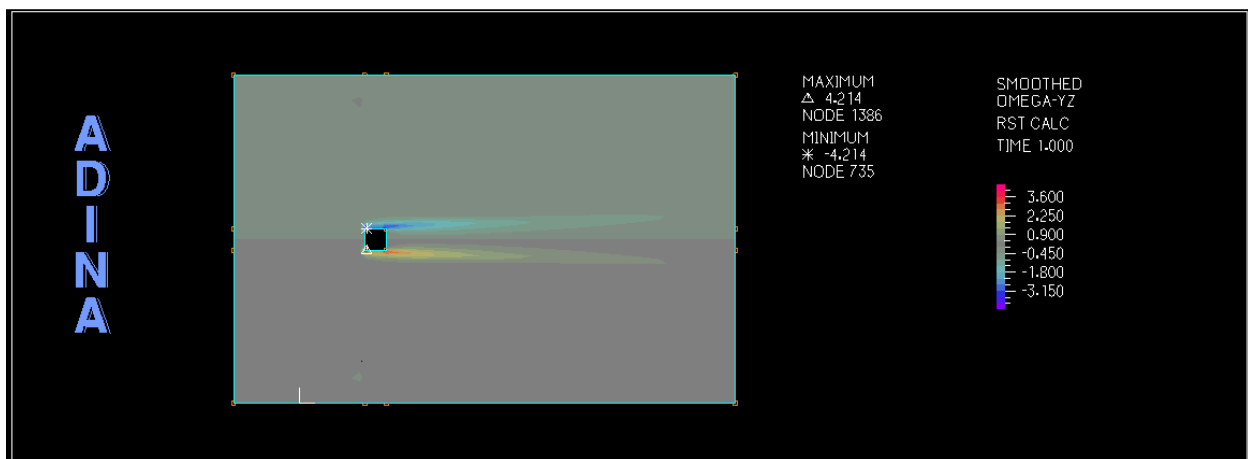


Figure 26: Omega-yz

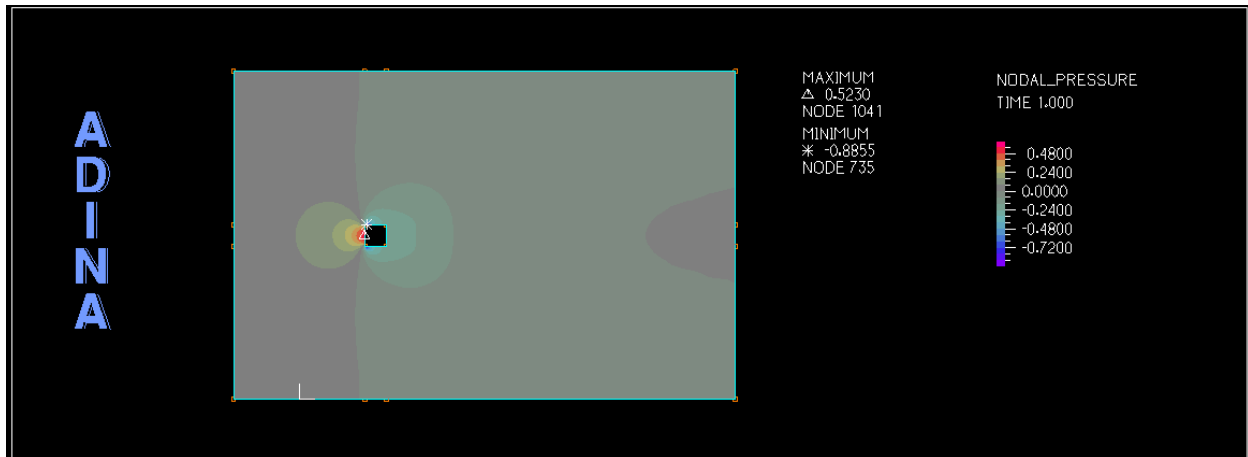


Figure 27: Nodal pressure

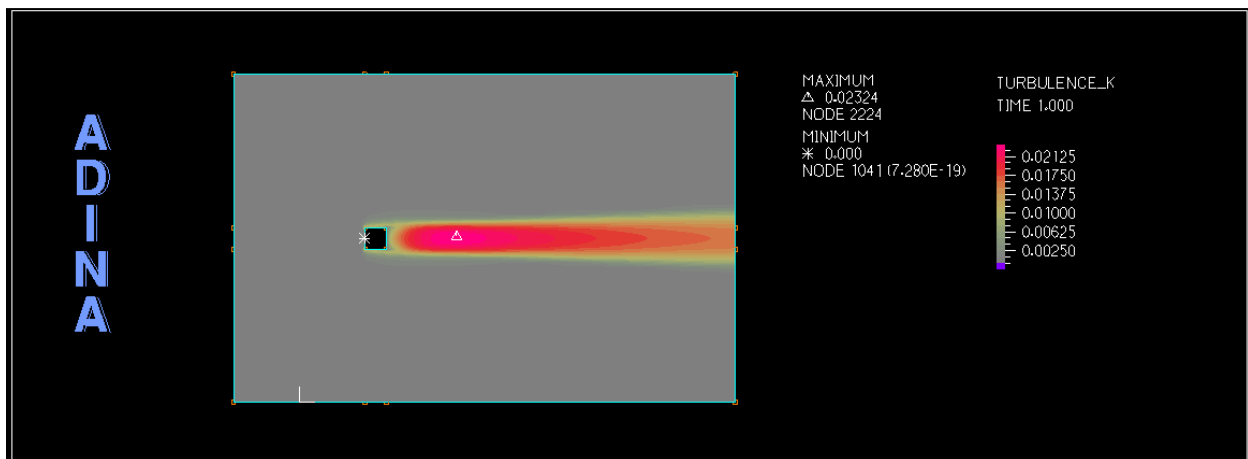


Figure 28: Turbulence kinetic energy

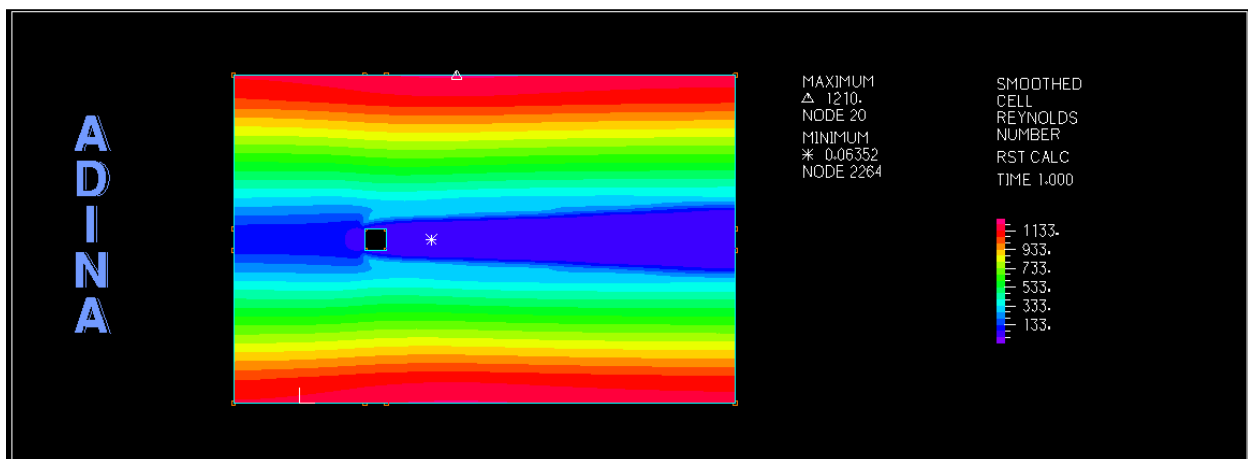


Figure 29: Cell Reynolds number

## 4. Conclusion

The current status of the available turbulence models in the finite element software like ADINA makes it possible to evaluate the flow past bluff bodies. This analysis can be then applied during the design phase to improve the aerodynamics of the structure and reduce the forces. Also the effect of circulation and vortices may be studied and effectively dealt with. With the advancement in the computational methods application of finite element in the field of computational wind engineering will surely increase.

## 5. References

1. S. Murakami, Computational wind engineering, *Journal of Wind Engineering and Industrial Aerodynamic* **36** (1990), pp. 517–538.
2. S. Murakami, Comparison of various turbulence models applied to a bluff body, *Journal of Wind Engineering and Industrial Aerodynamic* **46/47** (1993), pp. 21–36.
3. S. Murakami and A. Mochida, On turbulent vortex shedding flow past 2D square cylinder predicted by CFD, *Journal of Wind Engineering and Industrial Aerodynamic* **54/55** (1995), pp. 191–211
4. Klaus-Jürgen Bathe, *Finite Element Procedures*, Prentice Hall, 1996

MIT OpenCourseWare  
<http://ocw.mit.edu>

2.094 Finite Element Analysis of Solids and Fluids  
Spring 2011

For information about citing these materials or our Terms of Use, visit: <http://ocw.mit.edu/terms>.